



# Making PC Boards

Written By: Steven Robert Cypherd



## PARTS:

- [PCB etching kit \(1\)](#)

## SUMMARY

Making your own printed circuit boards (PCBs) is so easy now. The biggest change is in the resist coating on the boards. Now it is less sensitive to ambient light. You do not need a safelight or a closed room to work with these new boards. You do have to be careful with direct light from any source for more than a minute or so. You do not need a special bulb to expose them. Sunlight or a fluorescent light works best.

Plan your space. I closed my bedroom blinds early in the afternoon and placed the exposer frame under my desk. I put a towel over my bathroom window where I have the development tray filled and set up. That's it.

The DATAK Boards use the same developer. They require a Hot 100 – 110 degree water development and a Cold water rinse to fix the resist. They can be exposed with standard 100 watt bulb at 12 inches for about 10 – 15 minutes.

One thing that is important is that the resist is now sensitive to heat. Any heat. Storing a board in its package under direct sunlight will kill your board. Keep them in a cool dark place. Heat is a good thing too. When you are done developing your board you just rinse the board under warm water and that fixes the resist. Caution here. Once the resist is fixed it is

not sensitive to the developer or the light anymore. At this point 91% isopropyl alcohol cleans off the resist and you can fix errors or do modifications all in normal light.

One downside to these single fluorescent tube exposer frames is that the area of good light is just three inches wide down the length of the tube. A 4x6 board set at the front of the exposer frame will have its back one inch poorly exposed. You must rotate your board. For big boards just move them three inches each step through the exposer. Expose for about 6-8 minutes per step. You can get larger multi-tube exposer frames and special exposer lights.

[Jameco](#) has everything to make a PCB at home: positive resist boards, positive developer, etchant, trays and an exposer frame. Look under **Electronic Design → Prototyping Systems**.

Pre-sensitized boards are available in many standard sizes. The **More Products** category has PCB kits and stuff. I have a single 17" fluorescent tube exposer frame. Mine is about 5 ½" high. I got it at Fry's in Burbank, Ca. I also got my Ammonium Persulphate etchant at Fry's. Read the instructions that come with the boards. A good exposure makes a good PCB.

I use [Fritzing.org](#) to design my boards. Fritzing's software lets you export your design to a PDF file. Print the PDF file at 100% on clear transparency sheets. For laser printers print on the smooth side of the transparency sheet. Cut out the circuit on the board's outline.

To work with Fritzing you need electronics knowledge and desktop publishing experience. An example is the object label used in all views. Right-click the object and select **Show Label**. Select the object and left-click the label and drag it. Right-click the label, select **Edit, Size, Rotation and Display Details**.

You must get the breadboard and the schematic perfect before you can get a good PCB. No "birdcage wires" (Fritzing calls them "rat's nest" wires) anywhere. Test all circuit paths by clicking on a node. Then in the PCB remove any rat's nest wires, rotate and move the parts and delete and make new circuit paths. You make big changes in the breadboard view first, then the schematic view if you need it and then the PCB view.

In the PCB view deleting a trace is the same as deleting a wire in the breadboard and the schematic views. Complete your changes in the PCB and check for rat's nest wires in the other views. Make wires out of the correct connections and delete the rest. The default PCB is double-sided. Auto-routing is automatic in the other views as you add parts and wires. Start moving, orienting and wiring each part on your PCB as you add them.

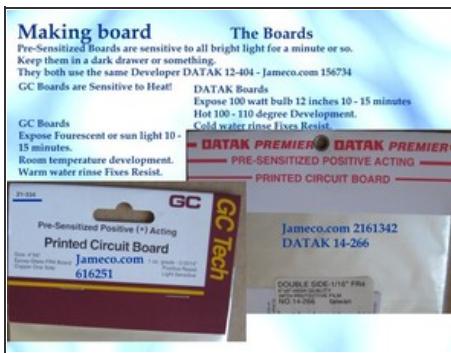
To make multiple circuits on one PCB, save your project as somethingA10. In PCB view

click the PCB and in the Parts menu set Sticky to unchecked. Click and drag the PCB away from your circuit. Set your PCB to the size that you need. 4x6 is width 101.6mm and height 152.4mm. Position your circuit to the left of the PCB and close to where the duplicate will go.

Drag a selection box around your circuit. Let go. In the edit menu select Duplicate (Ctrl-D). Wait for the duplicate circuit to show up. Do not click it. Use the arrow keys to move your duplicate onto your PCB. Repeat this for each copy. The breadboard and schematic views will be a mess. Line things up nicely. Save your project. Make an editable PDF. Wait for it. Done.

I do not put Sticky back on. Watch out for Duplicate errors. All of the part labels have copy numbers in them. Do not touch the breadboard or the schematic. To fix errors delete the bad circuit and make a new duplicate circuit. The more parts in the circuit the longer it will take to make a duplicate.

## Step 1 — Making PC Boards



- The DATAK Boards require a Hot 100 – 110 degree water development and a Cold water rinse to fix the resist.
- Supplies

## Step 2

**Step 2 Exposing the board**

If you have a good exposed board. You can take the protective film off both sides of the board and tape the top and bottom art work onto the board. Put a tight tinfoil box on the bottom. Block all light. Edges Too. Block the edges. Protect it and then expose the bottom.

Once your art work is ready Close the blinds and cover up direct light. Put your developer in its tray. Set the timer or the side exposor time. Remove the protective film on the resist side of the board. Place the art work onto the resist side of the board. Make sure it is oriented correctly. You can read the writing on the board. sure it is placed correctly. You can read the writing on the board. I use two pieces of class and photo clips as my art frame. You need to keep the wires tight to hold the board to get a good image. Place your art frame under the exposer frame a little in front of the frame. Mark the front edge if you need to turn the board. Turn on the exposer light and start the timer. Put the art frame in the sun. It is 10 minutes a side for smaller boards and 15 minutes a side for bigger boards. My board was 9 minutes each side. Rotated once. Double sides boards are 15 minutes each side. Turn over your board. When the timer for a side is up rotate your art frame to the next side. When the exposor time is over turn off the light or get out of the sun. Remove your board from the art frame. Protect the board from the light. Now is good time to put on your gloves.

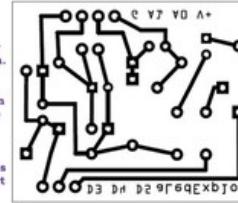
### Making PCBs

#### Printed Circuit Board Art Work

Any type of art work can make a PCB. This is my circuit board from Fritzing.com as a PDF for the bottom circuit pattern. Note: the lettering is backwards.

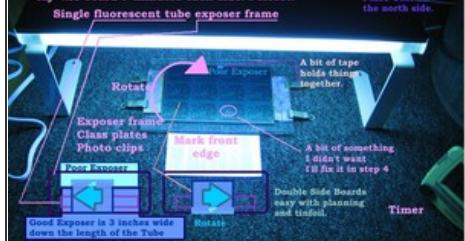
When you place this art on a PCB to be exposed make sure you can read the lettering or the image is correct.

You can play with Sharpies and paint to make your art work or circuit. Difference thickness will expose differently. This may cause an effect in etching.



### Making Boards Exposer

Correct exposor makes a good PCB  
10 minutes for small boards  
6 - 10 minutes a side for bigger boards  
Rotated to each side for exposer  
My 4x6 board 9 minutes each side. Perfect!



## ● Art Work

## ● Exposer

## Step 3

### Step 3 Development

Put your etchant into its tray and into a bigger tray with about a half inch of hot water in it. I use my 13x9 baking dish.

Remove the art work from your board. Place your board resist side up into the developer. Start an up counter on your timer. With fresh developer you should see the art work in two minutes. Keep agitating the tray. I rub the board with my gloved fingers. You can use a sponge too. Wait until all of the copper is shiny. About five minutes. All of the little holes need to be shiny too. If it is a double sided board keep flipping it over to check your progress. It is difficult to over develop your board. The time will increase after each board. The developer gets darker. If everything is good then rinse your board under warm water. This fixes the resist so it is not sensitive to light or the developer anymore. Open the blinds and turn on the lights. Pour the developer back into its bottle.

### Step 4 Fixing errors or making modifications

Using a paper towel wrapped toothpick and 91% Isopropyl Alcohol to clean out errors in the resist. Just dampen the corner with the alcohol. You do not want to mess up other areas of the resist. Add modifications. They have resist pens and art work.

### Making Boards

#### Fixing Errors



## ● Developing

## ● Fixing an error

## Step 4

### Step 5 Etching

It helps to warm up your etchant  
Place your board into the etchant copper side up  
Start an up counter on your timer  
Keep agitating the tray  
If it is a double sided board keep flipping it over to check your progress  
Change the hot water every fifteen minutes or so  
It can take up to half an hour to etch a board  
Once the etching starts it can move very fast. Check your board every 15 for the first half hour. Then every 10 minutes to the end.  
**You can over etch your board**  
When you are done etching rinse your board thoroughly  
Pour your etchant back into its bottle  
Rinse your trays out  
Rinse your tongs and everything else.

### Step 6 Cutting up your board

Dremel circular saws are big enough to keep a straight line easily  
Dremel has a circular saw adapter. I found it difficult to use.  
The blade kept binding in the PCB.  
Any fine saw will do  
Metal shears work too

### Double Sided Boards

My tip for Double Sided boards is using a thin wire like wire wrap wire. Drill your holes bigger about .030 or big enough to fit your lead easily. Put the thin wire in the hole and carefully solder it to the top trace. Try not to get solder into the hole. Push a lead through the hole to flatten the wire to the sides of the hole. For more current use up to four wires.

## Making Boards

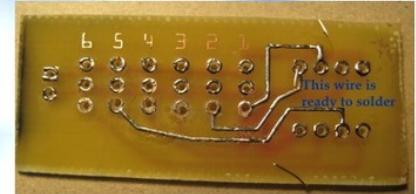
### Double Sided Boards

I use a small wire like a wire-wrap wire to connect the two sides. Drill your holes big enough to fit your lead easily. About .039in. Put the small wire through the hole and solder it to the top trace. Do not get too much solder in the hole.

Push one of the leads through the hole and press the small wire to the sides of the hole.

With square leads you can get about four small wires in.

Put your lead through the hole and bend and trim the small wire to fit the bottom trace and solder them.



## ● Etching

This document was last generated on 2012-12-15 03:50:51 AM.